



The 7th International Conference on Applied Energy – ICAE2015

CFD Ansys - Fluent simulation of prevention of dioxins formation via controlling homogeneous mass and heat transfer within circulated fluidized bed combustor

Eman Tora, Erik Dahlquist *

Future Energy Research Centre, Department of Energy, Building and Environment,
Mälardalen University, 721 23 Västerås, Sweden

Abstract

Internal mass and heat distribution within a combustor is a key factor of successful combustion process as it influences the contact between the reacting materials and helps reduce formation of undesired products such as dioxins via diminishing emerged hot or cold spots. Nevertheless in case of circulated fluidized bed combustor CFBC, it is difficult to keep a homogeneous heat and mass distribution due to the usage of combusting materials with variable and in some situations unpredictable compositions such as municipal and agricultural wastes. Thus this work investigates CFD Ansys – Fluent simulation as a tool to help the plant operator attain uniform heat and mass transfer via enabling the plant operator to adjust the operating conditions to fit well the used fuel. That is through: (a) visualizing the internal mass and heat distribution within a combustor, (b) manipulating the operating conditions of pressure, velocity, and bed particulates flow rate to determine the correct values considering the inevitable feature of the used fuels, (c) exploring operation deficits and sorting out the problems and (d) studying the feasibility of proposed modifications or changes. An illustrative case study is given as a methodology to demonstrate how likely reasons beyond combustor operation deficits can be defined and tackled. The case study is designed to exemplify the tool to achieve the objective of internal uniform heat and mass transfer within a combustor for given fuel composition and the operating conditions.

© 2015 Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (<http://creativecommons.org/licenses/by-nc-nd/4.0/>).

Peer-review under responsibility of Applied Energy Innovation Institute

Keywords: Circulated fluidized bed combustor; heat and mass transfer; CFD Ansys-Fluent simulation

* Corresponding author. Tel.: +46 - 073-960-7191; fax: +46- 016-15 36 00.

E-mail address: erik.dahlquist@mdh.se

1. Introduction

Circulated fluidized bed combustion technology gains a growing attention and application increase due to the ability to use different solid materials as fuel such as MSW and the agricultural residuals [1]; mostly these fuels have variable compositions and collected from different sources. Thus operating conditions of CFBC need to be regulated continuously to attain the correct values corresponding for every moment composition. Combustion process within the CFBC takes place between fuel and air (oxidizing agent) in existence of sand (heat transfer media) or other inert material. This is the primary view of the process, yet deeper investigation indicates that fuel is not a real lump material because it consists of different components (e.g. C, N, ash). When the combusted materials include chlorinated compounds, dioxins form if the temperature drops below 800°C. Thus complex overlapped processes occur simultaneously within the combustor. To keep the system under control, each CFBC unit needs to be analyzed separately to count for the available fuels. Nevertheless, there is a standard range of operating variables that can be taken as a primary guide for the plant operator [2] [3]. Failing to control these variables leads to undesirable side reactions and products. To enable the plant operator to control the plant operation in favor of getting uniform heat and mass transfer, CFD Ansys - Fluent is used to visualize the impact of the operating conditions. The simulation is conducted with the purpose of highlighting the capability of CFD to detect the problems that may encounter CFBC operator and then illustrate how to sort out the emerged problems; emphasis has been placed on avoiding dioxins formation via avoid cold spots formation.

2. Visualization, Monitoring and Control of CFBC Performance

To keep the system operating at the correct conditions an online control based on modelling of the system is needed. CFD, particularly Ansys-Fluent, is used herein, because of the high capacity to perform the relevant tasks, mentioned in the preceding section. To visualize CFBC core and process care must be given to the complex combustion occurring in a multiphase media with formation of many compounds and intermediates. Likewise, undesired processes may happen: erosion of the walls by the circulated particles - agglomeration and deposition of particulates which reduce the available heat transfer agent in the combustion unit - soot or/and melt of ash may form if the temperature degree and distribution wear out of control - eutectics may have a potential to form due to the existence of inorganic content such as K and Na in some materials. When combusting waste with high contents of Chlorine it is very important to have a high combustion temperature to avoid dioxin formation. If high moisture fuel is used with low mixing, cold zones may appear where dioxins can be formed, and these undesired products may follow passes through the boiler where they are not converted further. With modeling technique this can be studied. Complement will also be made by measuring the real temperature with a lance inside the boiler at different positions. This will be made during February, but was not done when the paper was written. However this study is devoted to the heat and mass transfer during the combustion process, the different processes have been brought out herein as this paper is an augmentation of a series of articles tackling modelling and optimization of CFBC via CFD.

3. Methodology

3.1. Case Study

CFBC unit with certain geometry adopted different open access resources is in operation to release heat from a solid fuel; this heat can be used to generate steam in the wall tubes to drive steam

turbines. The unit is working at atmospheric pressure under known conditions of flow rate and pressure, see Table 1. The CFBC flue gas analysis points to something wrong within the combustor as the flue gas has considerable mass fraction of C which means a poor combustion occurring. To figure out what goes wrong and afterwards sort out the problem, CFD Ansys-Fluent simulation is conducted using the same given geometry displayed in Fig. 1, and fuel composition and operating condition listed in Table 1.

3.2. Model and Simulation Set up

The simulated geometry is displayed in Fig. 1. The combustor consists of a tower and a cone. The tower has the dimensions of 28m H x 5m L x 10m W, and the cone has dimensions of 3.5m H x (5 m L top face, 2,5 m L bottom face) x 10m W. At the top of the tower on the right wall, there are two ports of the flue gas released to the cyclones. The cone right side has two ports of recycled sand return and nozzles of the secondary air entrance. On the left side of the cone, there are four ports of fuel injection and other ports of secondary air inlet. At the bottom of the cone there are the net of the primary air nozzles, and there are three pans to collect the settled ash.

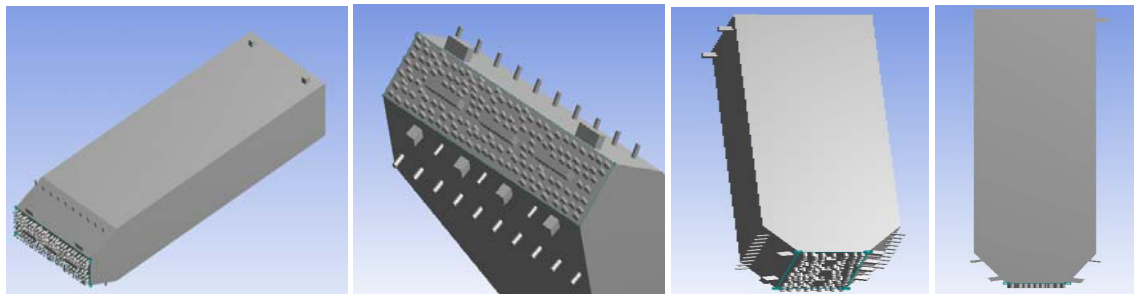


Fig. 1. Different views of the simulated geometry.

Table 1. Fuel composition and operating conditions

Zone	Boundary Conditions					
	Type	Value	Thermal	Species	Flow direction	DPM
Ash out	Pressure outlet	1 atm	300 K	0	Normal to boundary	escape
Fuel in	Mass flow rate composition (0.4 C, 0.03H, 0.07 S, 0.02 N, 0.11 O, 0.37 ash)	0.04 kg/s	300 K	1		reflect
Primary air Secair1/Secairr Secairl	<u>Given case:</u> pressure inlet <u>Modified operation:</u> velocity inlet	1 atm 5m/s	1100K	0		reflect
To cyclone	Pressure outlet	1 atm	1200 K	1		escape
Wall	Stationary, Aluminum	Temperature at 1200 K	NA	NA	NA	reflect

Steady state pressure-based solver is used with activating the gravity effect. The boundary conditions of each zone are given in Table 1. DPM stands for discrete phase model; secair refers to the secondary air. The turbulence flow is described by the realized k-epsilon model; internal radiation heat transfer is modeled using P1 model as it is fast and gives good results. The Non-Premixed Combustion Model available in Ansys – Fluent [4] [5] is selected to model the combustion within CFBC whereby air and fuel do not get mixed before entering the combustor. It is capable of taking the user fuel composition, and without going through the details of complex chemistry, it predicts the combustion produced compounds and intermediates. Here the details chemistry of dioxins formation have not been simulated,

as this study deals with the reasons beyond dioxins formation (cold spots formation) instead of the chemistry itself. Thus heat distribution and temperature level is the controlling factor.

Two simulations have been conducted: first represents the given case in order to visualize the problem seeking to define the reasons beyond it, second investigate the interfered problem reason to validate or refute it with the purpose of indentifying the needed changes or modifications to attain a homogeneous heat and mass transfer. As there divers processes take place, a set of models have used as shown in Fig. 2.

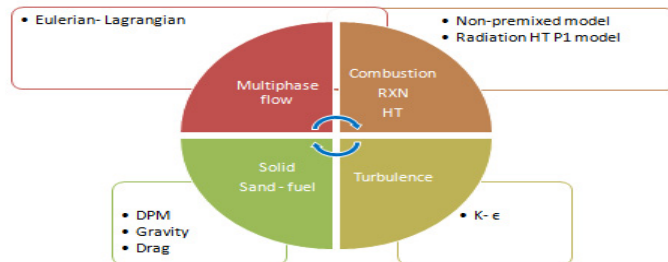


Fig. 2. Models used in the simulation

4. Results

4.1. Given case simulation

The simulation of the given case using Fig. 3 and table 1 displays that carbon inside the combustor has bad distribution and so there is no enough mixing between it and the air as shown in Fig. 4, which can be a reason of poor combustion. Regularly the reason of bad mixing may be low velocity, therefore the velocity contour has been developed, see Fig. 5. It is obvious how much the velocity is lower than the needed to circulate the bed and the reacting materials as according to ref. velocity needs to be in the range of 3.5 – 12 m/s. practically, within the given conditions of pressure, the combustor act as FB but not CFB.

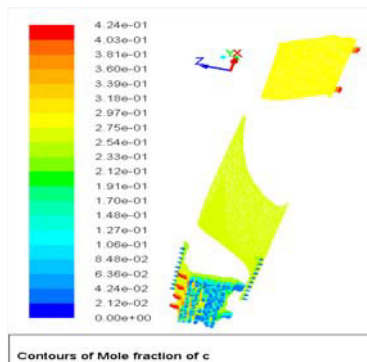


Fig. 3 Contours of C inside the combustor

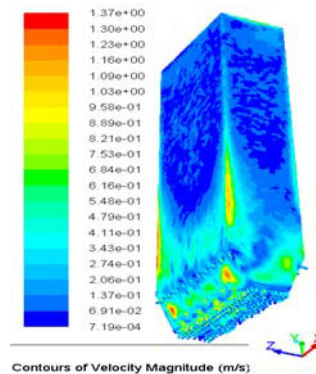


Fig. 4 Contour of velocity of the given case

Now the step is to track these deficits in order to fix the problem and that is what to be done in the next section.

4.2. Modified case simulation

According to the above section, it is interfered that the velocity was too low to circulate the particulate which reduces the mass and heat transfer leading to separate flow and poor combustion. To

validate that and to indicate the capability of simulation to predict the performance with manipulated operating conditions, the entering primary and secondary air has been set to a velocity of 5 m/s with keeping all the other operating conditions as it is. So any changes appear can be attributed to changing the velocity. Figure 5 shows the internal velocity streamlines, velocity range is 3 m/s – 9 m/s which means it is controlled within the proper range. Figure 6 shows the C contour. Comparing these two contours with the corresponding figures (Fig. 3 and Fig. 4) representing the original case implies the problem has been sorted out and now the combustor operates well. Most of carbon has been combusted near to the fuel ports which is represented by red areas, see Fig. 6. Moreover, there is a uniform temperature distribution and the temperature range at the optimum range, 850°C – 930°C. Table 2 (See Appendix A) lists the temperature degree at the different surfaces of the combustor and also the heat transfer rate and direction where (-) ve sign refers to heat release, but (+) ve means heat input or consumed. Figure 7 shows the internal heat distribution and it is clear that the temperature is controlled within the desired level, and even the green spot refers to the minimum temperature degree, above 800°C so dioxins cannot form at the operating conditions. That is attributed to the appropriate mass transfer reflected in the residence time of the particulates inside the combustor which enhances the heat transfer, see Fig. 8 where most of the circulated bed particulates have a residence time of 5 – 7 sec which is standard.

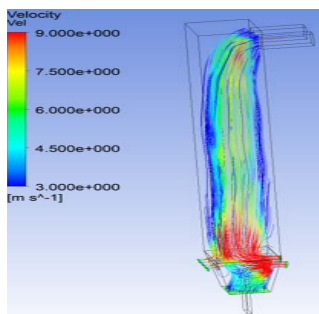


Fig. 5. Velocity streamlines of modified case

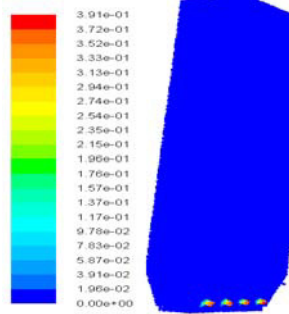


Fig. 6. Carbon contour

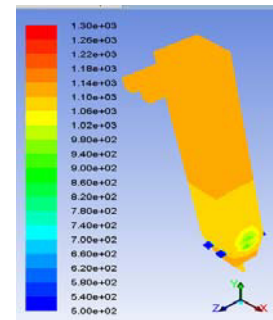


Fig. 7 Temperature contour

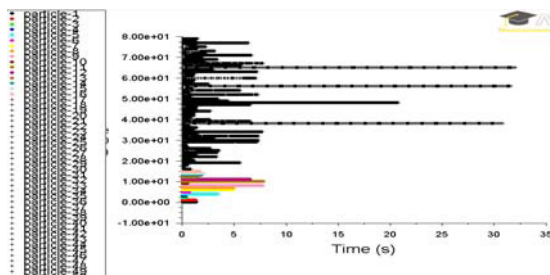
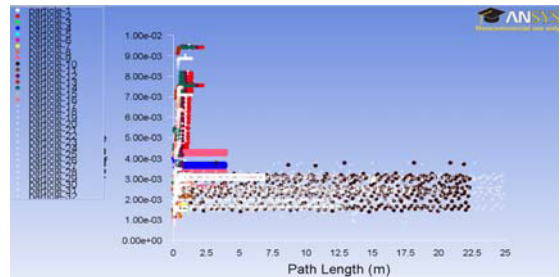


Fig.8. Residence time of sand particulates

Fig. 9. CO₂ path lines, each colour represents one particle of C converted to CO₂

Other indication of the combustion efficiency is the produced CO₂, see Fig. 9. CO₂ forms obviously around the fuel ports. This indicates the importance of optimizing the fuel ports which will be a topic of a separate study.

Part of the ash exits with the flue gas and another part drops to combustor bottom where collected in pans after exchanging its heat content with the primary air as a preheating step. The simulation succeeds to

predict both; mass balance of the ash as estimated by Ansys is listed in Table 3, see Appendix A. It specifies the ash content of the fuel stream: part of it exits with the flue gas (tocyclone face), another part dropped to the bottom of the combustor (ashout face), and there is a net amount stayed inside the combustor as part of the circulated bed.

5. Conclusion

- CFD Ansys –Fluent simulation has been used to determine the reasons for poor combustion and then point to the likely improvement routes which have been validated via the simulation results.
- Simulation results indicate that the fuel distribution between the ports should be adjusted to make the temperature and velocity profiles in the corners more equal, especially when the fuel has higher moisture content.
- When the fuel is very wet and also content of PVC is significant it should be compensated by mixing in dryer fuel.
- Herein several variables have been considered; however, this study is counted here as an opening for next simulations with more details with widening the simulation in terms of the scale and associated processes.
- Simulation verification is going on based on experimental results currently taking place at our laboratories in conjunction with Malarenergy power plant of Västerås City, Sweden.

Copyright

Authors keep full copyright over papers published in Energy Procedia

References

- [1] JEA Large-Scale CFB Demonstration Project, Detailed Public Design Report, USA Department of Energy, National Energy Technology Laboratory (NETL), Pittsburgh, Pennsylvania 15236, Cooperative Agreement No. DE-FC21-90MC27403, June 2003
- [2] United Nation Environment Programs; <http://www.uneptie.org/energy>; accessed on 31/01/2015
- [3] Northampton Circulating Fluidized Bed Power Plant;
www.power-technology.com/projects/northampton/; accessed on 31/01/2015
- [4] Ansys – Fluent User's Guide, available at http://www.arc.vt.edu/ansys_help/flu_ug/flu_ug.html; accessed on 31/01/2015
- [5] Ansys Workbench User's Guide; available at
http://orange.engr.ucdavis.edu/Documentation12.1/121/wb2_help.pdf

Biography

Erik Dahlquist is a professor of Energy Technology and Research Director of the School of Business, Society & Engineering. Got BSc from Uppsala University 1974 in Chemistry and Biology, PhD at KTH in 1991. Adjunct professor at KTH, Royal inst of Technology 1997-2000. Full professor at Mälardalen University, Västerås, from 2000.

Appendix A

(i) Heat Distribution within the combustor

Table 2: Heat generation

Total Heat Transfer Rate (w)		Mass-Weighted Average Total Temperature (K)	
ashout	-446067.72	secair1	578.85901
fuelin	1361856.9	fuelin	379.60797
limein	156.92017	interior-solid	1226.2804
primaryairin	-539706.75	secair2	1103.2533
recycledsand	-298635.75	primair	1242.9927
secainin	-16305.94	recycledsand	1176.7256
secairin	-16248.742	secair3	1107.5338
tocyclone	-627443.5	secair4	1106.8239
wall-solid	198478.19	tocyclone	1216.275
		Net	1226.246

(ii) Ash formation and distribution

Table 3. Mass-weighted average of ash

Zone	Ash value	Zone	Ash Value	Zone		Ash Value	Zone	Ash Value
fuelin	0.36469656	interior-solid	0.11830676	secair		0	to cyclone	0.1184094
ashout	0.0069137616	primaryairin	0	recycled sand		0.0027601584	Net	0.11830648